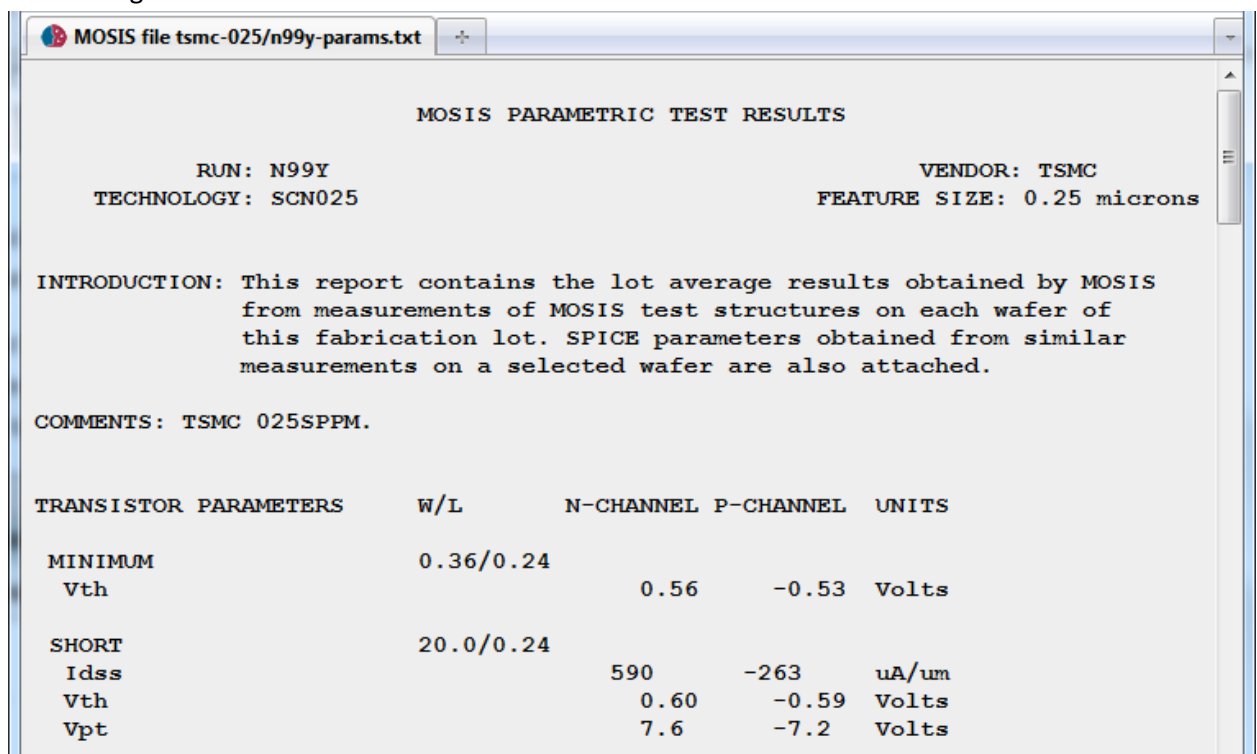


Using TSMC Transistor Models from MOSIS in LT Spice

This is a quick start guide on how to use the MOSIS Wafer Electrical Test Data and SPICE Model Parameters to create MOSFET models for LT Spice simulation. For LT Spice downloads and tutorials the reader is referred to <http://www.engr.sjsu.edu/ebasham>.

1. Wafer Test Data is available for MOSIS processes at <http://www.mosis.com/test/>
We will be using the TSMC 0.25 μ m N99Y test data for simulation. The download screen should look like figure 1.



The screenshot shows a web browser window with the title "MOSIS file tsmc-025/n99y-params.txt". The main content is titled "MOSIS PARAMETRIC TEST RESULTS" and contains the following text:

RUN: N99Y
TECHNOLOGY: SCN025

VENDOR: TSMC
FEATURE SIZE: 0.25 microns

INTRODUCTION: This report contains the lot average results obtained by MOSIS from measurements of MOSIS test structures on each wafer of this fabrication lot. SPICE parameters obtained from similar measurements on a selected wafer are also attached.

COMMENTS: TSMC 025SPPM.

TRANSISTOR PARAMETERS	W/L	N-CHANNEL	P-CHANNEL	UNITS
MINIMUM Vth	0.36/0.24	0.56	-0.53	Volts
SHORT Idss	20.0/0.24	590	-263	uA/um
Vth		0.60	-0.59	Volts
Vpt		7.6	-7.2	Volts

Figure 1. Website for Wafer Test Data from <http://www.mosis.com/cgi-bin/cgiwrap/umosis/swp/params/tsmc-025/n99y-params.txt>

2. Scroll to the bottom of the page and select the NMOS and PMOS model data, shown in figure 2.

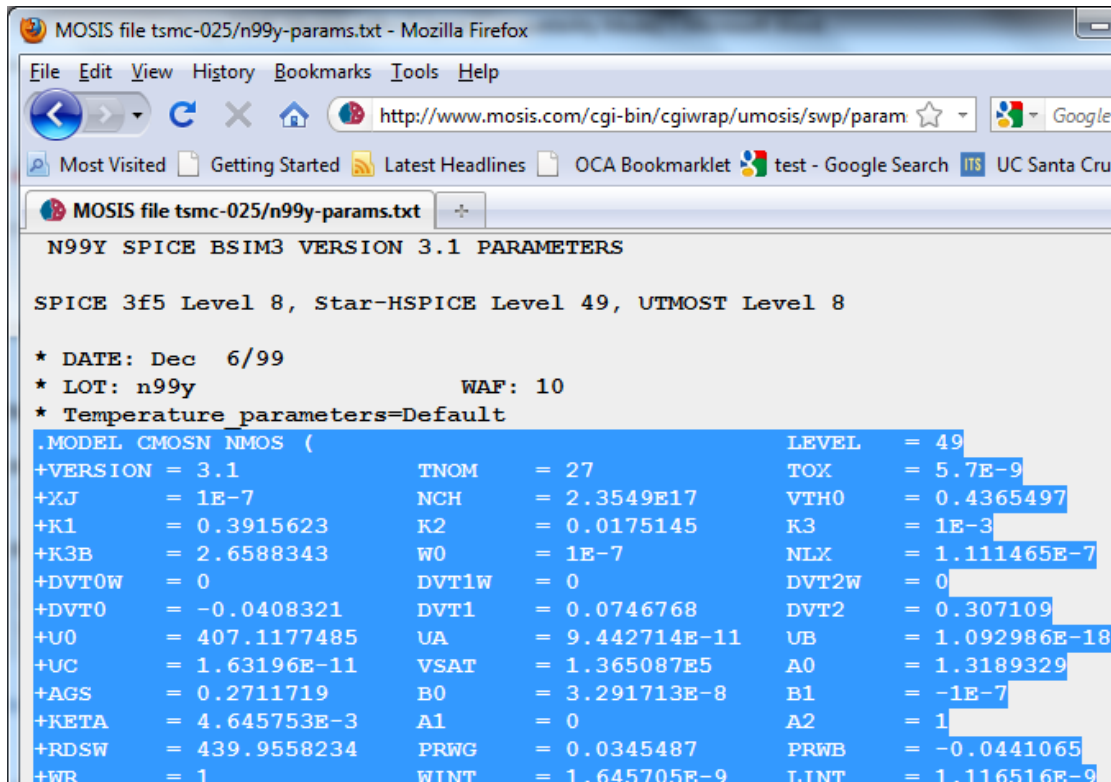


Figure 2. Model data selected. The NMOS model is shown, but the file contains both nmos and pmos models.

3. Copy and paste this data into text file called TSMC_models.txt. Edit the file so the first line of each transistor model file reads as follows:

```
.model      tsmc25n      nmos      LEVEL = 49
.model      tsmc25p      pmos      LEVEL = 49
```

4. Place the text file in the SAME directory as your LT Spice simulation files.
5. Open a new schematic and place an include command by clicking the “.op” button, shown in figure 3.

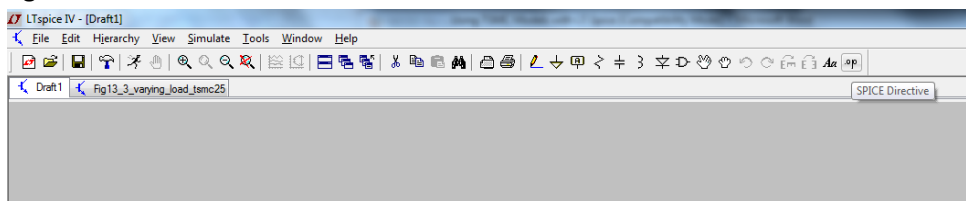


Figure 3. Inserting a spice directive.

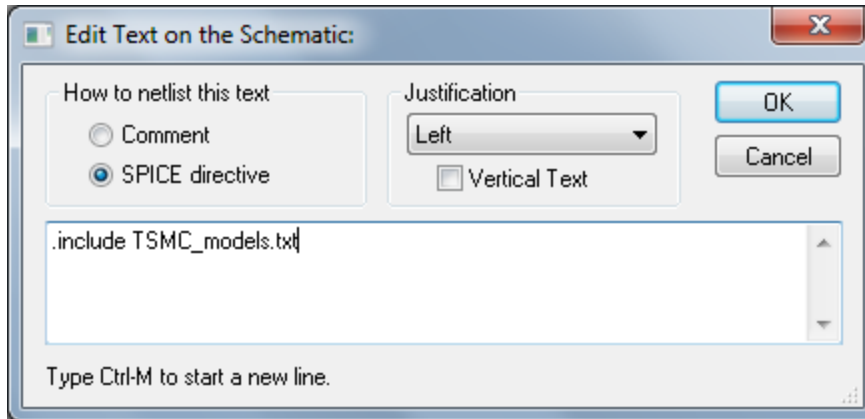


Figure 4. The `.include TSMC_models.txt` command

6. Select the component button.

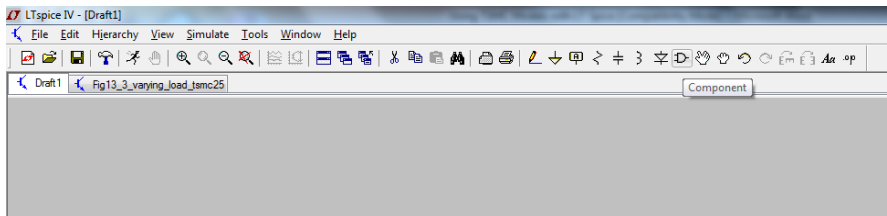


Figure 5. Component selection number.

7. Select the “nmos4” device type, and place it on the schematic. (figure 6).

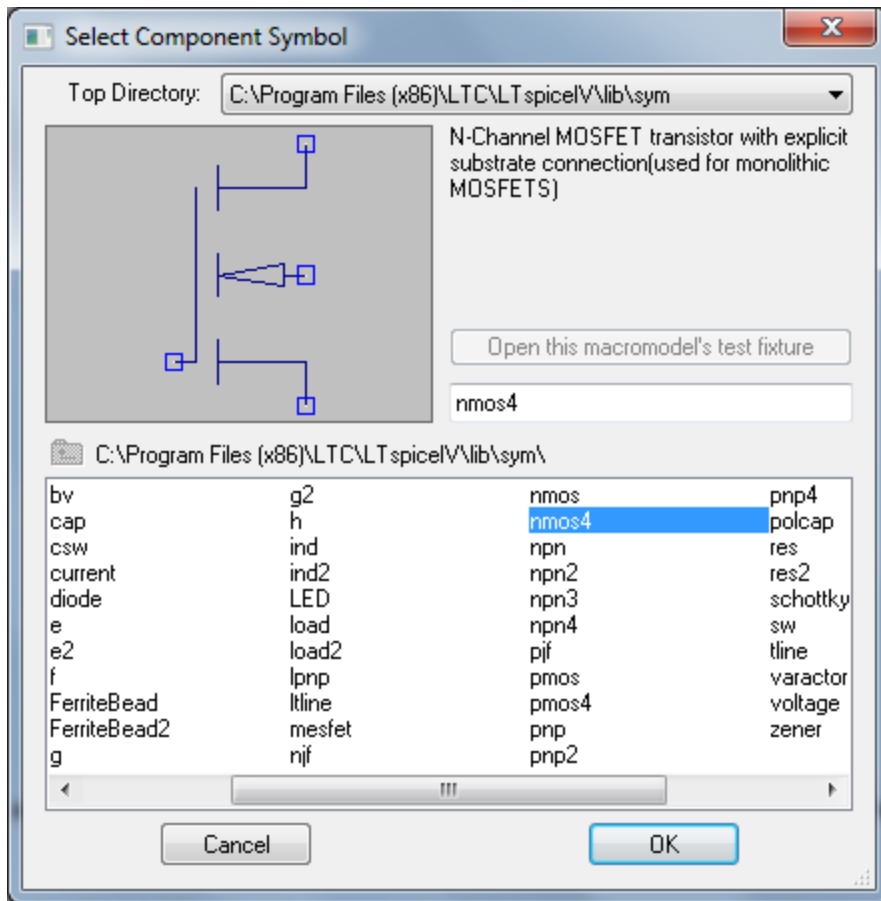


Figure 6. Transistor edit dialog box

- Right click on the transistor to bring up the monolithic mosfet edit dialog box.

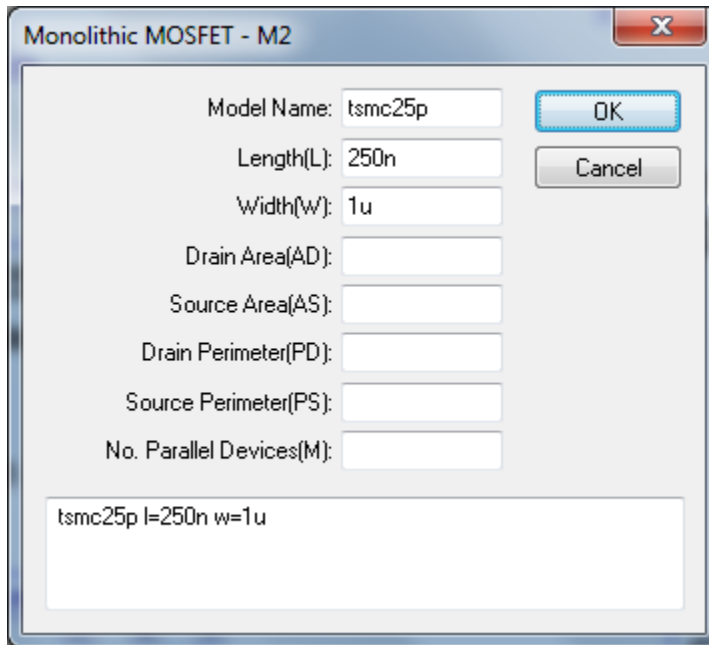


Figure 7. Editing the monolithic MOSFET

- If you would like to display the width and length on the schematic, you will have to right click on the device name "TSMC25n" and fill out the dialog as in figure 8.

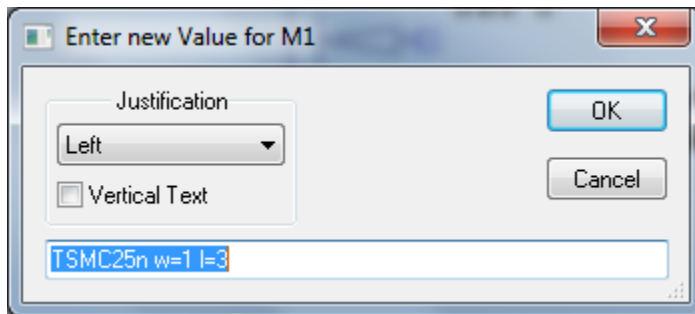


Figure 8. Right clicking brings up the edit dialog.